

Understanding Columbia's Reentry Problems

Thomas Paul. Lockheed Martin Space Operations, Houston, TX

(Updated with NASA comments/corrections)

Soon after the Space Shuttle Columbia accident occurred last year, a group of CFD analysts from NASA centers and private industry was organized to help determine the cause of the accident. This group was under the direction of the Applied Aeroscience and CFD Branch of the Aeroscience and Flight Mechanics Division at the Johnson Space Center. For external flow simulations, noncommercial CFD codes that specialize in hypersonic or high Mach number flows were used. These tools were used to determine heating rates, pressures, and temperatures for a large number of vehicle damage scenarios. Lockheed Martin Space Operations was called upon to provide CFD support in the area of internal flows within the shuttle wing cavity, and for these simulations, FLUENT 6.1 was chosen. Two large-scale, simplified models were run to understand the flow patterns once a breach of the internal wing cavity was initiated. The results were primarily used to visualize flow patterns within the wing cavity.

The first CFD model included the entire left wing without the wheel well cavity. The purpose of the first model, which did not include the reinforced carbon-carbon (RCC) cavity along the wing leading edge, was to visualize the flow field within the wing cavity immediately after the leading edge spar breach. This model assumed that the flow coming into the wing cavity was normal to the spar. It included all of the primary vents that allow for flow between the main cavities of the wing. A six-inch diameter hole was modeled in the spar at the approximate location where the spar breach was judged to have occurred, which was between RCC panels 8 and 9. The results of the modeling showed that at this location, the high temperature, high velocity gas stream entering the wing cavity impinged on the outboard wheel well cavity. Instrumentation in the Shuttle wheel well cavity registered abnormal temperatures during reentry, so the FLUENT results helped support the conclusion of the accident investigation team that the spar breach was in the RCC panel 8-9 area, and that the initial spar breach was likely entering the wing cavity normal to the spar. This model also showed that the flow entering the wing cavity tended to swirl within the cavity just outboard of the wheel well, and did not initially penetrate further into the rear cavities of the wing. The second CFD model was a 2-D simulation of the left wing cavity and the RCC cavity. It was used to visualize the flow through the RCC breach, through the wing spar breach, and into the wing cavity directly outboard of the wheel well. The purpose of this model was to verify whether or not it was possible for the flow to come into the wing cavity normal to the leading edge spar. The results from this 2-D model showed that the internal structure behind the RCC panels probably deflected the flow entering the RCC cavity so that it impinged normal to the spar. As in the 3-D model, this deflected flow stream was found to impinge on the wheel well outer wall. This model again supported the conclusions regarding the location of the spar breach and how the flow behaved inside the wing cavity.

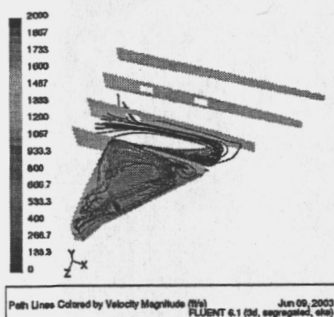
OPTIONAL FORM 99 (7-90)

FAX TRANSMITTAL		# of pages
To <i>Denise Romero</i>	From <i>Tom Paul</i>	1
Dept / Agency	Phone # <i>281-333-6770</i>	
Fax # <i>281-483-2965</i>	Fax # <i>281-333-7727</i>	
NSN 7540-01-317 7308	5098-101	GENERAL SERVICES ADMINISTRATION

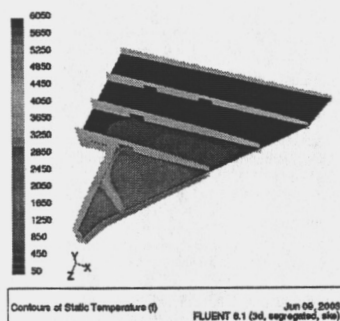
Reference

1. Columbia Accident Investigation Board Final Report, Volume 5, Book 3, part 13, section 5.3.6.3, "CFD of Simplified Internal Wing Geometry", pg. 467-480.

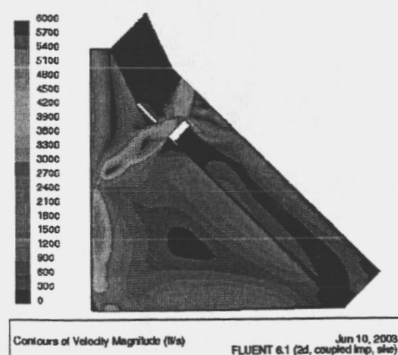
Figures:



shuttle-path.tif: Pathlines inside the wing cavity, showing impingement on the wheel well wall at left by the jet entering through a breach in the wing



shuttle-temp.tif: Temperature contours show the heating caused on the wheel well wall by the hot, high speed jet entering through the wing breach



shuttle-2d-vel.tif: 2D simulation showing the path of the jet entering through the RCC breach on the outside of the vehicle, then through the spar breach, and into the wing cavity, ultimately hitting the wheel well wall

may use wing-geometry.tif as graphical element